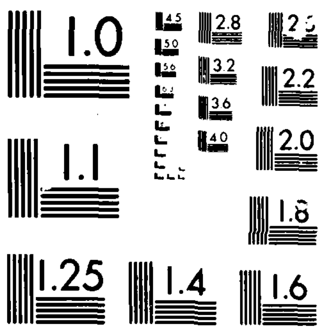
AD-A166 960 GAMM (GESELLSCHAFT FUER ANGEWANDTE MATHEMATIK UND MECHANIK) CONFERENCE ON.. (U) OFFICE OF NAVAL RESEARCH LONDON (ENGLAND) E F BROWN 01 APR 86 ONRL-C-1-86 1/1

UNCLASSIFIED F/G 20/4 NL



MICROCOP'

CHART

(12)

ONRL Report C-1-86

AD-A166 960

GAMM Conference on Numerical Methods
in Fluid Mechanics

Eugene F. Brown

April 1, 1986

DTIC FILE COPY

U.S. Office of Naval Research, London

UNCLASSIFIED
SECURITY CLASSIFICATION OF THIS PAGE

REPORT DOCUMENTATION PAGE

1a. REPORT SECURITY CLASSIFICATION UNCLASSIFIED	1b. RESTRICTIVE MARKINGS
2a. SECURITY CLASSIFICATION AUTHORITY	3. DISTRIBUTION / AVAILABILITY OF REPORT
2b. DECLASSIFICATION / DOWNGRADING SCHEDULE	
4. PERFORMING ORGANIZATION REPORT NUMBER(S) C-1-86	5. MONITORING ORGANIZATION REPORT NUMBER(S)
6a. NAME OF PERFORMING ORGANIZATION US Office of Naval Research Branch Office, London 6b. OFFICE SYMBOL (If applicable) ONRL	7a. NAME OF MONITORING ORGANIZATION
6c. ADDRESS (City, State, and ZIP Code) Box 39 FPO, NY 09510	7b. ADDRESS (City, State, and ZIP Code)
8a. NAME OF FUNDING / SPONSORING ORGANIZATION 8b. OFFICE SYMBOL (If applicable)	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER
8c. ADDRESS (City, State, and ZIP Code)	10. SOURCE OF FUNDING NUMBERS PROGRAM ELEMENT NO. / PROJECT NO. / TASK NO. / WORK UNIT ACCESSION NO.

11. TITLE (Include Security Classification)
GAMM Conference on Numerical Methods in Fluid Mechanics

12. PERSONAL AUTHOR(S)
Eugene F. Brown

13a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT (Year, Month, Day)	15. PAGE COUNT
Conference	FROM ____ TO ____	1 April 1986	5

16. SUPPLEMENTARY NOTATION

17. COSATI CODES

FIELD	GROUP	SUB-GROUP
20	04	

18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)
Fluid mechanics
Fluid dynamic properties

19. ABSTRACT (Continue on reverse if necessary and identify by block number)

Presentations as aero-acoustic modeling, transition control, vortex shedding, domain decomposition methods, TVD calculations, viscous/inviscid interaction, and Euler solutions are reviewed. Papers dealing with methods (in contrast with applications) were stressed at this meeting.

20. DISTRIBUTION / AVAILABILITY OF ABSTRACT ☒ UNCLASSIFIED/UNLIMITED ☐ SAME AS RPT. ☐ DTIC USERS	21. ABSTRACT SECURITY CLASSIFICATION UNCLASSIFIED
22a. NAME OF RESPONSIBLE INDIVIDUAL C.J. Fox	22b. TELEPHONE (Include Area Code) (44-1) 409-4340 22c. OFFICE SYMBOL 11

DD FORM 1473, 84 MAR — 83 APR edition may be used until exhausted. All other editions are obsolete.

SECURITY CLASSIFICATION OF THIS PAGE
UNCLASSIFIED

U.S. Government Printing Office: 1985-507-047

CONTENTS

GAMM CONFERENCE ON NUMERICAL METHODS IN FLUID MECHANICS

Introduction

The sixth Gesellschaft für angewandte Mathematik und Mechanik (GAMM) Conference on Numerical Methods in Fluid Mechanics was held from 25 through 27 September 1985 in Göttingen, West Germany. In attendance were approximately 200 scientists from Europe as well as Russia, India, Japan, and the US. The meeting was hosted by Deutsche Forschungs- und Versuchsanstalt für Luft- und Raumfahrt (DFVLR). Papers dealing with methods (in contrast with applications) were stressed in this meeting. During the meeting, papers on Galerkin, finite difference, finite volume, and hybrid methods for the solution of the Navier-Stokes and Euler equations were presented. Related to these matters, several papers dealing with grid generation, numerical dissipation, and the implementation of downstream boundary conditions were also presented. I will not be able to report on all of the papers which were presented because there were two parallel sessions in progress during approximately one-half of the meeting. A proceedings volume consisting of all papers presented at the conference will be published by Vieweg-Verlag in *Notes on Numerical Fluid Mechanics* and should be available shortly.

Aero-Acoustic Modeling

The first paper was by J.C. Franjaud and J.M. Hervouet from Electricité de France. Their interest was in silencing piping systems for carrying steam from the boilers to the generators in steam power plants. They needed a way to predict the aerodynamic noise produced in such systems. Initially, they set up a test in a transparent test rig in which hot-wire, acoustic, laser Doppler velocimetry (LDV), and interferometric measurements were made. The working fluid was air. Various components present in steam distribution systems could be inserted upstream of the test section. In the case of an orifice, they found that the flow leaving the orifice reattached to the upper side of the test section, producing a highly three-dimensional asymmetric condition.

In an attempt to simulate this test, they used a hydrodynamic code which they modified for compressibility effects. To simulate the turbulence, the k-ε turbulence model was used. A fractionary step integration approach was used in which the advection terms were integrated by a characteristic method. Initially, their calculations were carried out using a law-of-the-wall model. However, this model gave a symmetric rather than the desired asymmetric solution. When they refined the grid, discarded the law-of-the wall model, and carried out their calculations all the way to the wall, they found the asymmetric results which they had seen in their experiments. For both calculations they used the measured upstream velocity profile as inlet boundary conditions. This illustrates the importance of correctly modeling the wall boundary conditions.

A second calculation was made which involved the flow through a dual-throat nozzle. A two-dimensional Euler code was used which employed a fractionary-step integration approach, similar to that used in the hydrodynamics code. In order to accurately predict the acoustic features of the flow, a new turbulence model was developed. It included, in addition to the standard turbulence kinetic energy and turbulence dissipation quantities, transport equations for two new variables: the acoustic correlation and an energetic variable. Although a new acoustic-oriented turbulence model was developed, no results were available to validate its performance. Nevertheless, some careful thought seems to have gone into the construction of their turbulence model. Because of this, further developments of this research should be closely watched.

Transition Control

The second paper was given by E. Laurien, of the DFVLR, Göttingen. He described his spectral calculations of

the transition mechanisms responsible for the laminarizing effects of suction and blowing. In order to produce the onset of turbulence, he introduced spatially periodic disturbances. The evolution of these disturbances was modeled with the three-dimensional, time-dependent, Navier-Stokes equations in a coordinate system which moved downstream at the speed of the Tollmien-Schlichting waves. What his calculations showed was that blowing and suction produced a significant decrease in the turbulence fluctuations compared with the growth predicted when no boundary-layer control was present. In addition, the simulated effects of blowing and suction on the production of turbulent fluctuations agreed well with experimental measurements. Both the well-known harmonic and subharmonic transition structures were predicted by the code and it was possible to extract from the model the details of the flow mechanisms responsible for the transition phenomenon. The calculations showed that transition can be delayed if suction or blowing is provided early enough in the transition process. This was substantiated by shear maps, which were clearly devoid of the tongue-like, three-dimensional structures which are known precursors of turbulence. This is not to say that the effect of the disturbances had been removed. Span-wise vortices were still present; however, they did not develop into the types of structures which are known to lead to turbulence. This indicates that transition control, while delaying transition, certainly cannot return the flow to its initial uniform state.

Vortex Shedding

P. Kiehm of the Institute for Thermal and Fluid Dynamics at the Ruhr University in Bochum, West Germany, described his use of a marker and cell (MAC) technique to calculate the flow around a confined circular cylinder. Configurations where the cylinder was both confined by planes perpendicular and parallel to the axis of the cylinder were examined. Upstream of the cylinder the velocity profile was assumed to be parabolic. Of particular interest was the Reynolds number at which periodic vortex shedding was observed. It was found that confining the cylinder produced a significant increase in the Reynolds number at which periodic shedding was observed; for high blockage ratios, Reynolds numbers as high as 67 were found compared with Reynolds numbers of 40 for an unconfined cylinder. Also, confinement was found to damp out the wake flow downstream of the cylinder and return the velocity profiles to their initial parabolic form more quickly than in the unconfined case. In addition, for the axially confined cylinder, large velocity components in the direction of the cylinder axis were predicted. This contributed to a strong helical motion which occurred downstream of the cylinder, see Figure 1.

Domain Decomposition Methods

W. Fritz (Dornier) talked about his multiblock approach for calculating two-dimensional flow around complex geometries. The concept here was to subdivide the flow field into a number of topologically simple subregions to simplify the grid generation process. Both inviscid and boundary-layer-corrected inviscid calculations were shown. One of the calculations which he showed was the

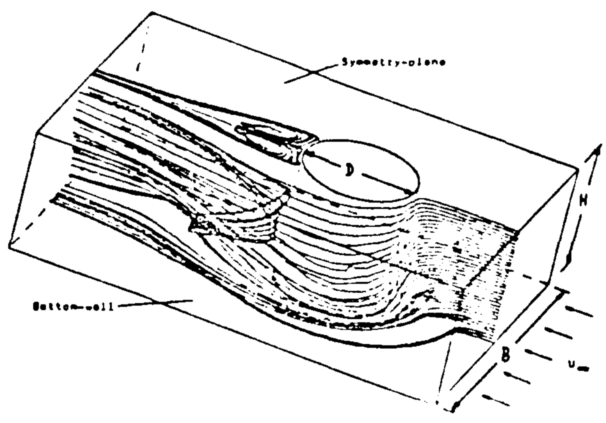


Figure 1. Confined circular cylinder.

flow over an automobile. In this case the Reynolds number ranged from 1×10^6 to 5×10^6. Both the viscous and boundary-layer-corrected calculations predicted separated flow at the rear of the body. The phenomenon responsible for the separation was completely different in the two cases, however. In the case of the viscous calculations, the separation was due to the existence of an unfavorable pressure gradient at the rear of the automobile. This resulted in the separation point being located upstream of the base of the body and produced a value of the base pressure which was in reasonable agreement with experiment. In the inviscid calculations, on the other hand, the zone of separation was restricted to the base region itself and was due to the sharp corners at the edge of the base region. These corners produced a total pressure loss (even though the calculations were inviscid) because of the discretization errors produced by the rapidly changing metrics. Not unexpectedly, the base pressure predicated by the inviscid calculations was in poor agreement with experiment.

By far the most sophisticated application of the multiblock approach was presented by E. Venkatapathy (NASA, Ames). Instead of the term multiblock he preferred to use "patched grids," and he defined several ways in which patching could take place. The advantages of patched (multiblock) grids are that they allow good alignment between the grid and the flow and a fine mesh can be produced exactly where it is needed. On the other hand, difficulties are introduced by this approach, primarily because of the increased complexity of the data structure. This is because with overlapped grids there is a need for both global and local indexing and the computed data must be passed between the global and local meshes in an efficient fashion. The numerical algorithm which was used in the calculations was the conservative supra characteristic method (CSCM), which is a diagonally dominant, approximate-factorization technique. It has the important characteristic that it is an upwind scheme. For patching methods, upwinding is an absolute requirement. Linear interpolation was used to exchange data between the grid systems.

The fine mesh of the local grid has less numerical diffusion and thus produces much sharper shock calculations. Venkatapathy showed a calculation of Mach 5.0 flow over a cone-cylinder which clearly demonstrated the improvements which can be obtained with patched grids. A comparison was shown between a coarse (26×26) grid on which patching was employed and a fine grid (101×101) in which no patching was used. The results with patching on the coarse grid were at least as good as the fine mesh calculations. The patching technique can be automated, thus producing an evolutionary, adaptive grid structure which adjusts to the complexities of the flow as they are encountered in the calculations.

A particularly interesting application of the patching approach was presented by M. Rai of the Information General Corporation, Mountain View, California. He showed a rotor-stator interaction problem in which the rotor grid moved relative to the stator grid. In his presentation, he highlighted the treatment of the interface points. He indicated that interface procedures must be stable, accurate in space and time, conservative, applicable to generalized coordinate systems, and easy to incorporate in existing codes. For geometric simplicity, he used circular-arc, biconvex airfoils for the rotor and stator blades. Identical H-grids were used for both the stationary (stator) and the moving (rotor) patches. Osher's total variation diminishing (TVD) scheme was used to solve the unsteady Euler equations. A spectacular color motion picture showing the case of a 1.5 Mach number inlet flow illustrated the results of his calculations. The interaction between the rotor and the stator flow fields was dramatically evident. The resolution of the calculations seemed excellent, particularly in the vicinity of the shock-shock interactions. A segment showing a "zoomed-in" view of the flow at the leading edge of the stator

plates clearly showed the periodic attachment and the detachment of the shock in response to the flow generated by the rotor. There was no visible evidence of the patching interface in any of the calculations. In addition to turbomachinery problems, this method can be applied to helicopter blade flows, propfan calculations, and propeller/fuselage interaction problems.

TVD Calculations

H.C. Yee (NASA, Ames) presented a new symmetric-differencing, total variation diminishing (TVD) scheme and compared it with the original upwind TVD scheme developed by Osher. Her calculations showed that a 30 to 40 percent savings in computational time was possible with the new method.

In another TVD calculation, C.M. Kwong (University of Salford, UK) highlighted an important advantage of TVD methods; namely, that shocks can be captured without the spurious oscillations which often occur when the MacCormack algorithm is used. Kwong showed calculations of flow in a shock tube, flow in a channel with a bump, and an oblique shock reflection calculation in which such oscillations were notably absent. Because TVD schemes offer a rational method for controlling the proper amount of dissipation, no problem-dependent adjustments were needed to achieve this result.

Viscous/Inviscid Interaction

M. Schmatz, Messerschmitt-Bölkow-Blohm (MBB), presented his strong viscous/inviscid interaction airfoil calculations. His approach was to divide the flow field into three zones: a zone upstream but far away from the airfoil where the Euler equations are used, a region upstream but near the airfoil where a displacement-thickness-corrected Euler calculation is used, and a region downstream of the airfoil where the Navier-Stokes equations are used. This is shown in Figure 2. The equivalent inviscid flow which results from the Euler/boundary-layer calculations is iteratively coupled with the Navier-

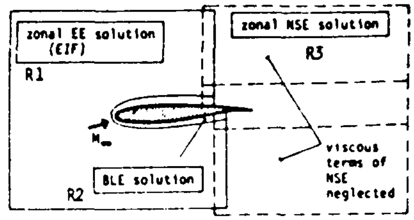


Figure 2. Overlapping subdomains.

Stokes solution by means of a modified Schmatz alternating algorithm. A Baldwin-Lomax algebraic two-layer stress model was used for the turbulence calculations. Results were shown at Mach numbers of 0.504, 0.72, and 0.83 at three angles of attack and at Reynolds numbers of approximately 6×10^6. Some difficulties were encountered in accurately predicting the skin friction, and wiggles appeared downstream of the region where the Euler and Navier-Stokes codes were coupled. To remove these wiggles it was suggested that an implicit Euler code or characteristic-oriented Navier-Stokes code could be used. The big advantage of Schmatz's method lies in three-dimensional calculations and not in the two-dimensional cases which were presented here. Future plans include extending this work to two-dimensional separated flows and then to three-dimensional calculations.

D. Chaussee (NASA, Ames) presented his parabolized Navier-Stokes calculation for the flow over a cone. A Baldwin-Lomax turbulence model was used. Shock waves were fitted using characteristic theory to reduce the number of points needed in the calculations. Calculations at 4-, 10-, and 14-degree angles of attack were made with as many as 121×45 points in the cross plane. Even with the finest grid it was impossible to obtain grid-insensitive solutions, especially on the pressure surface of the cone. Accordingly, a run with 10^7 points was made. This calculation required approximately 2.2 hours on a CRAY X-MP.

Euler Solutions

L. Eriksson of the Aeronautical Research Institute of Sweden (FFA) showed how it was possible with local grid refinement to greatly increase the accuracy of the calculation without greatly increasing the number of computational points. First, a coarse mesh solution was obtained, then, in the region selected for local grid refinement, values of the flow variables were estimated either by linear means or by means of extrapolation of the appropriate Riemann invariants. In a calculation made on a NASA 0012 airfoil, 25 percent more points were added in the vicinity of the nose. The surprising result was that this reduced the total pressure error by 75 percent. In addition, grid refinement produced a sharpened shock structure which in some cases persisted even beyond the zone selected for grid refinement.

A. Rizzi (FFA) described his inviscid calculations of the flow over a cranked, twisted, delta wing. The purpose of his calculations was to study unstable flows. Such calculations are only possible if there are modes in such flows which do not experience explosive growth. For these calculations an explicit, finite-volume technique was used employing second- and fourth-order artificial dissipation. Runge-Kutta integration was used. Calculations were presented for a Mach number of 1.2 at an angle of attack of 10 degrees. Both medium mesh (80,000 points) and fine mesh (640,000 points) calculations were made; however, it was not possible to obtain a mesh-independent solution. Based on the level of artificial viscosity which was present in the flow, the probable value of the Reynolds number was approximately 1,000.

Summary

At the end of the meeting the session chairmen were invited to give their reactions to the papers which had been presented and to summarize for the audience their perceptions of the state of the art in computational fluid dynamics. They generally agreed that three-dimensional Euler simulations involving upwards of 250,000 points were commonplace, but whether or not these codes presently exist in versions which could be used for production calculations is a matter of opinion. However, within the next decade the necessary degree of user friendliness for production applications will certainly be developed. The outlook for Navier-Stokes codes is less clear. Production codes employing simple turbulence models should be available within the next decade. These codes will enable three-dimensional calculations involving more than eight million points to be accomplished and will most certainly make use of large eddy simulations.

Other chairmen expressed a more conservative viewpoint and stressed the need for new algorithms (for both vector and parallel computations) for inversion of large matrices. They expressed the need for more accurate Euler schemes to minimize the effects of numerical dissipation. They felt that, for some time to come, multizone decomposition employing some type of adaptive scheme will be an alternative to Navier-Stokes calculations for practical computations.

Whatever the code, attention must be given to reducing its development time. There must be better reporting of one another's mistakes and of the strength and weaknesses of methods now in existence.

A final comment was made by M. Napolitano of the University of Bari, Italy, who said that although inflationary claims of code capability needed to be avoided, inflationary code utilization should not. "You need to be critical of a code" he said, "but not too cautious." Often, he said, a new approach is discovered by examining the failures of a calculation which is pushed too far. He cited, as an example, the failure of conventional boundary layer techniques which when applied to separated flows led to the development of indirect boundary-layer methods.

END
FILMED

6-86

DTIC